SIEMENS NX
HELMET EXERCISE GUIDE

Version(s) supported: NX 12.X

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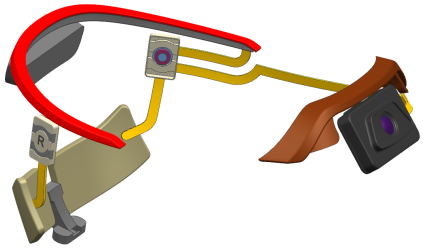
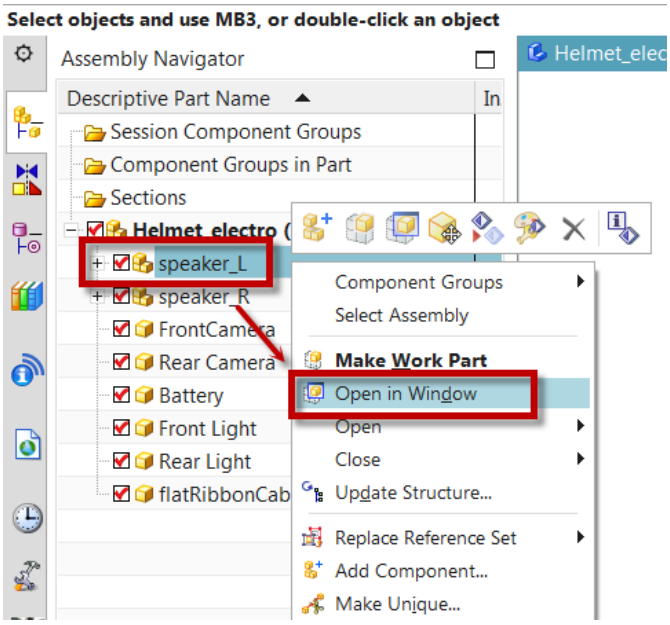
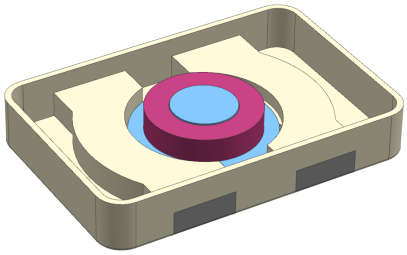
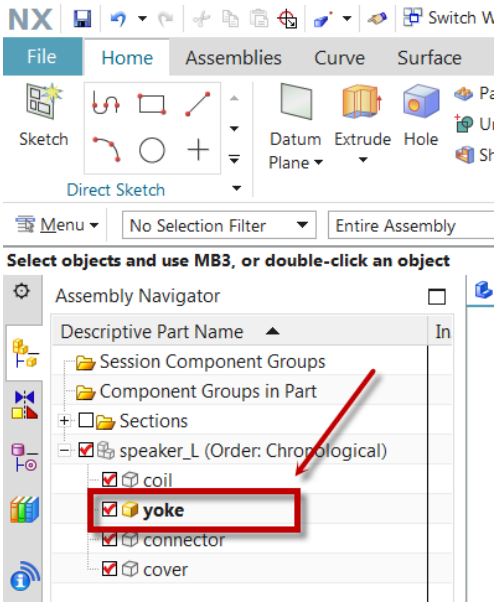
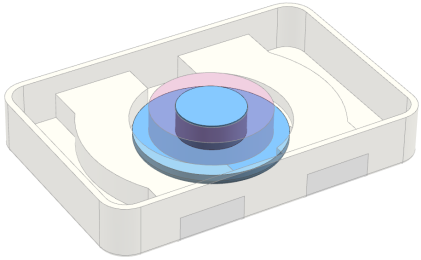
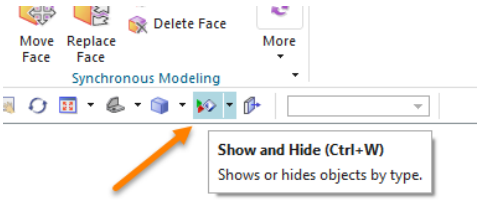
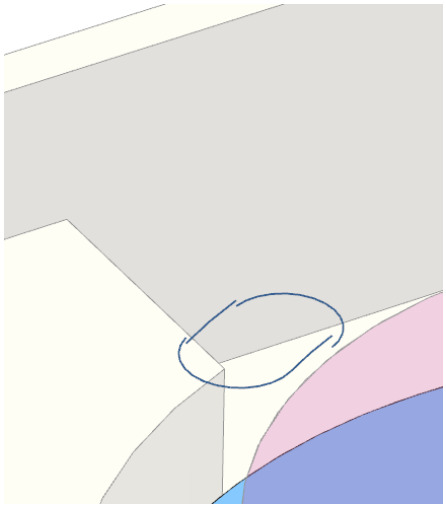
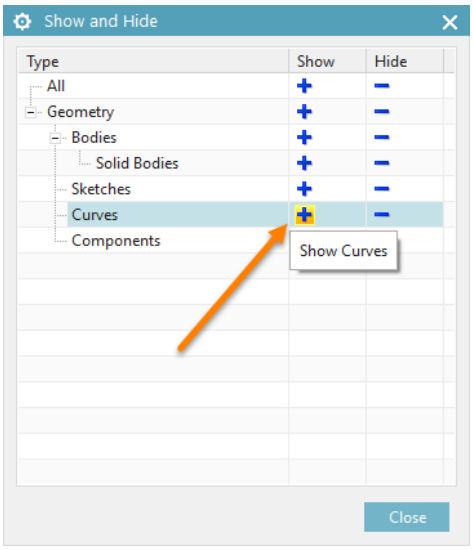
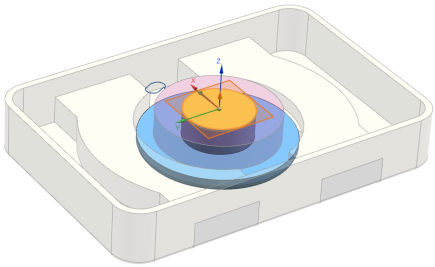
Sketching in NX using 2D Synchronous Technology

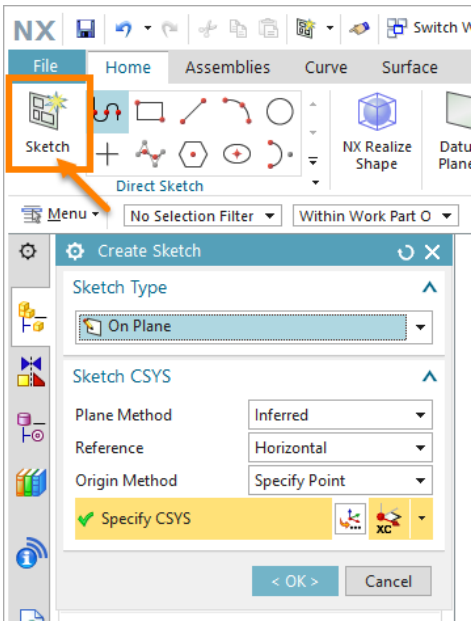
Introduction

Synchronous Technology in NX makes sketching and modeling fast and intuitive and gives the user the freedom to modify geometry using simple push-and-pull methods. This tutorial shows you how to work with an imported sketch, make modifications to the sketch and use the sketch for adding features to the geometry.

1	To start, open C://data_sketch/Helmet_electro.prt The model used for this tutorial is “Helmet_electro.prt”	
2	In the **Assembly Navigator**, select sub-assembly “speaker_L” → right-mouse button → select ‘**Open in Window’** 	

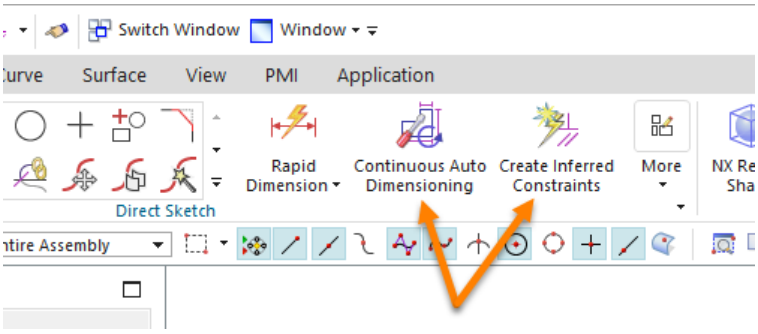
3	In the **Assembly Navigator** → double-click → " yoke" to make it the **Work Part** 	
4	Go to the **Top Border** bar → click **Show and Hide** → Click **+** → Click **Close** 	

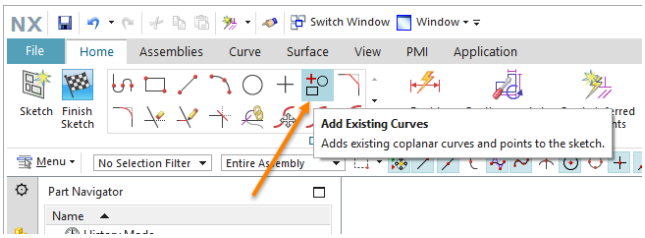
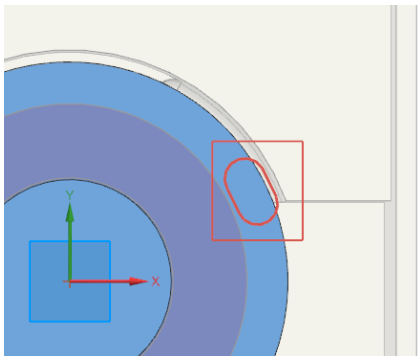
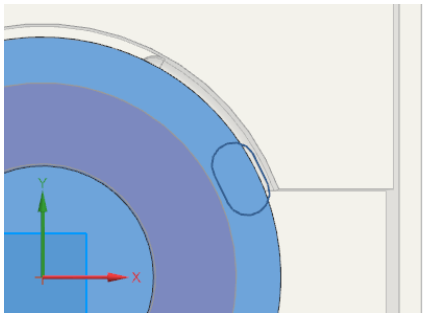
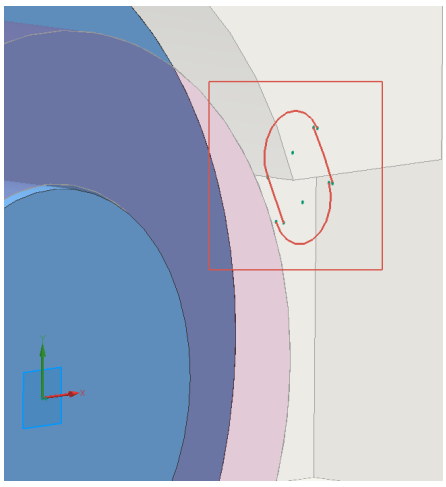
		
Note: *These are imported curves that have some errors and do not have any constraints. It highlights a common problem where translated geometry will have integrity issues, such as gaps, overlaps, non-tangencies, or curves that should be on the same plane but are not. Synchronous Technology 2D in NX enables users to quickly fix these issues, as well as make changes to these curves without the need for constraints.*		
5	Go to the **Home** tab and click **Sketch**.	

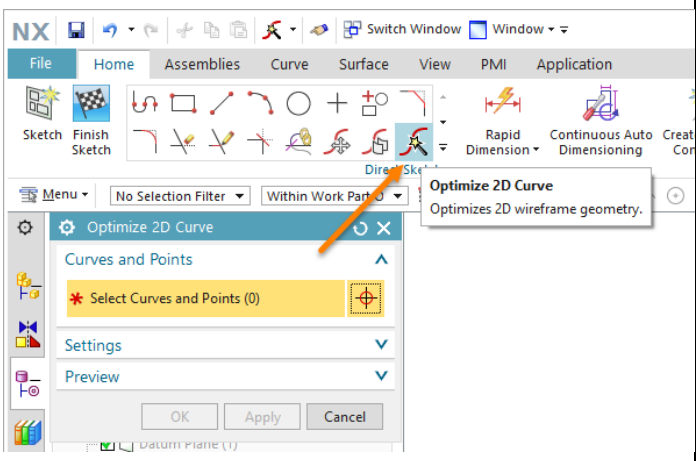


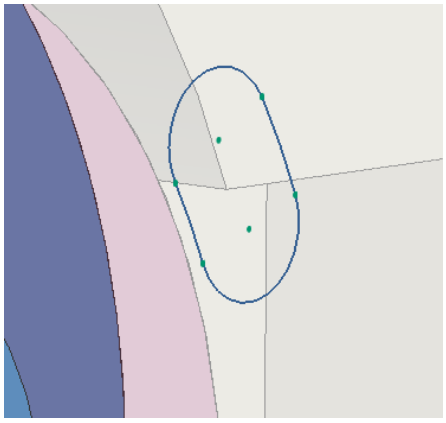
Select the top planar face of the coil as sketch plane → click **OK** (close the dialog)

Note: *Turn off "Create inferred constrains" and "Continuous Auto Dimensioning" from the **Home** tab. Since we will be using Synchronous Technology 2D to make changes to this profile, we don't want to deal with constraints, since they are not necessary and will save us quite a bit of time.*



6	Choose **Add Existing Curves** from the **Home** tab  Select the imported curves → by dragging a box across the entire profile → Click **OK**	 
Note: *The Optimize 2D Curve command cleans up and simplifies imported geometry and points. For example, it will eliminate duplicate curves, close gaps (snap curves to their endpoints), simplify curve types (convert B-curves into analytic), move all curves to the sketch plane, and so on. It is a very valuable tool for quickly cleaning up imported and legacy data.*		
7	On the **Home** tab → click **Optimize 2D Curve**	



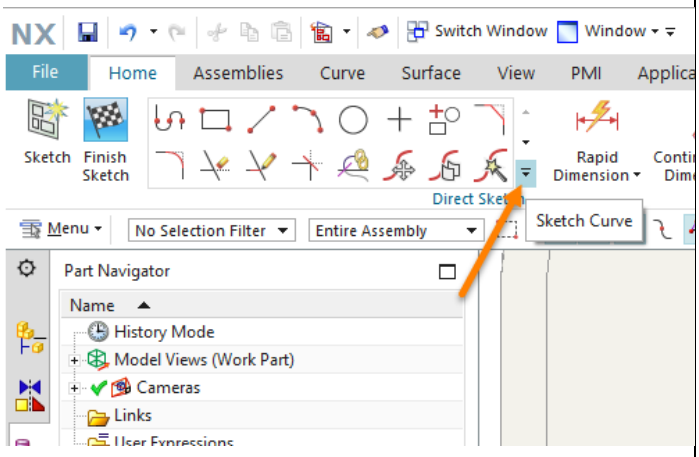


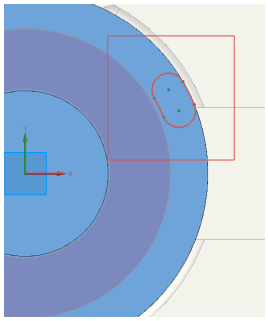
Drag a box → select all curves → click **OK**

Note: *Hold down the middle mouse buton to rotate the model, if needed. On the keyboard, use Shift+F8 keys to orient the view back to the plane of the sketch.*

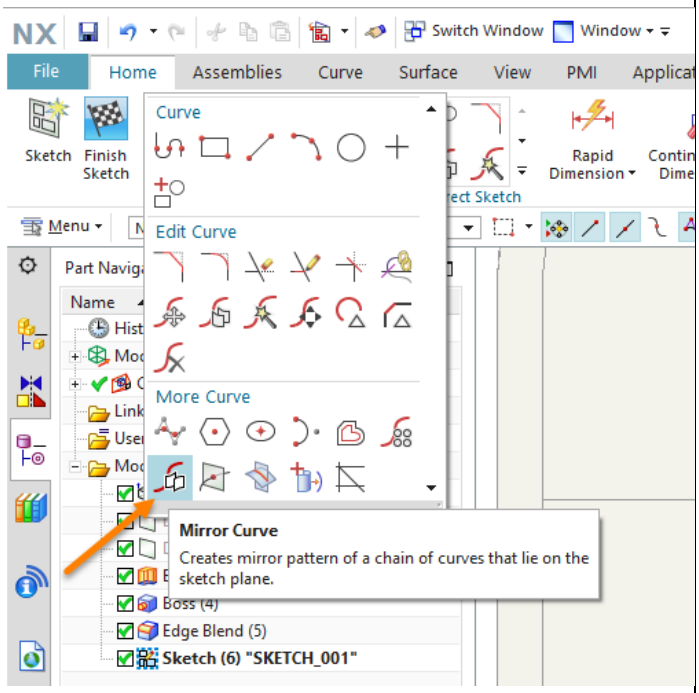
The Top Border bar has many filtering, selection and view manipulation options.

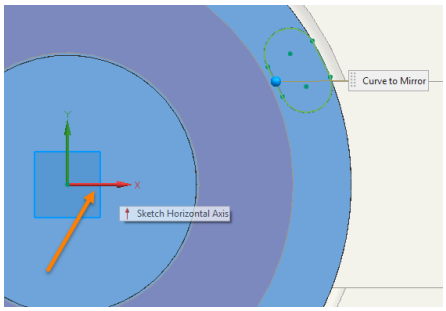
8 Click **Expand** button → see the rest of the **Sketch Curve** commands

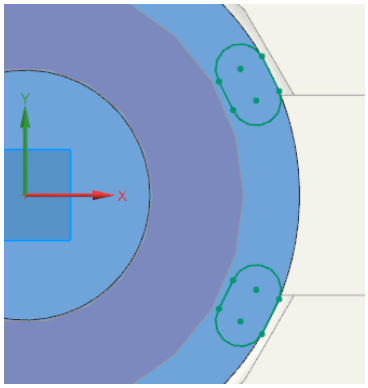




Choose the **Mirror Curve** command



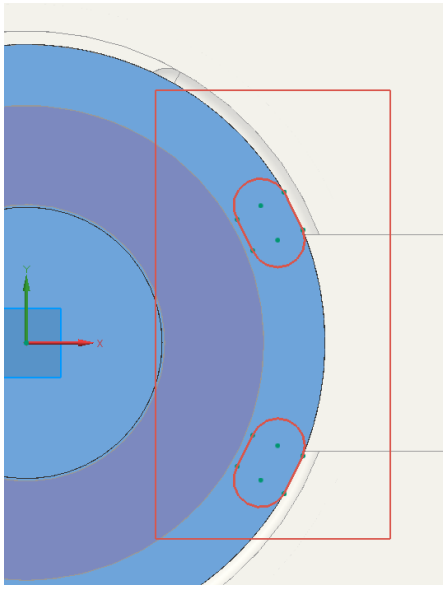
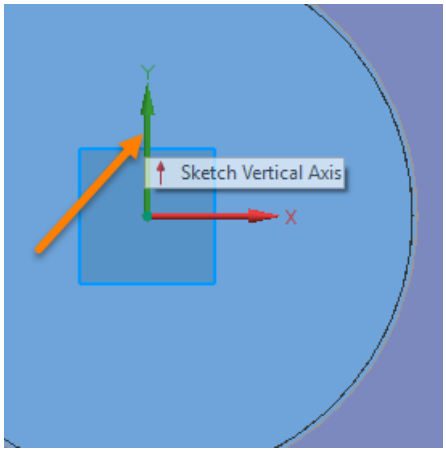
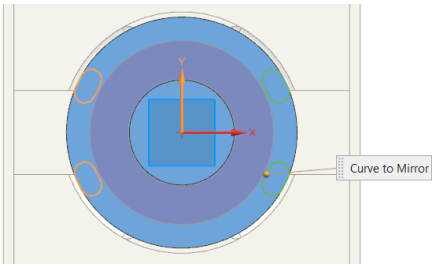




Drag a box to select all curves.

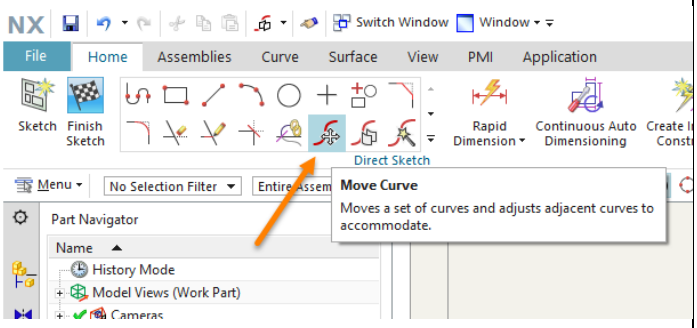
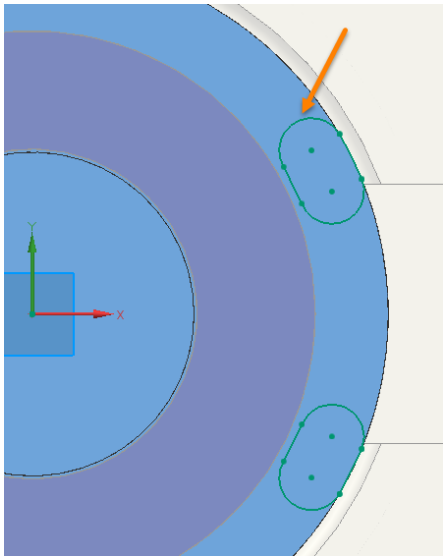
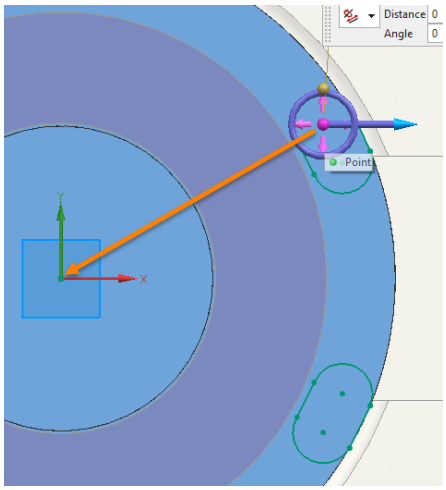
In the **Mirror Curve** dialogue window → click on **Select Centerline** → Click on the **X** axis (in the graphics window) → click **Apply**

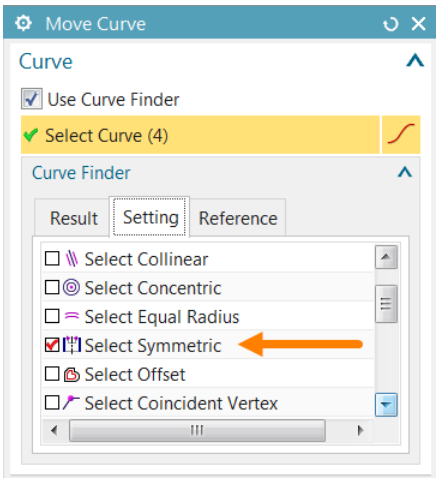
Keep the **Mirror Curve** dialogue window open

9	Repeat the above procedure → select both profiles by dragging a box to mirror them about the **Y** axis → Click **OK**	  

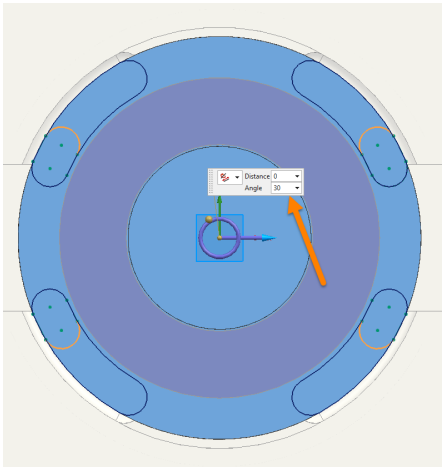
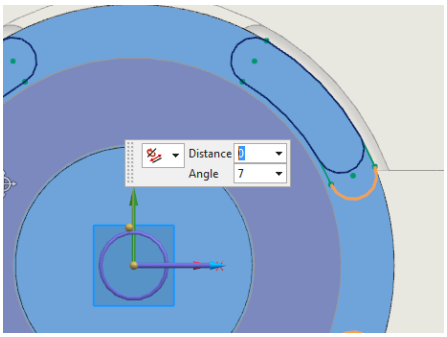
***Note**: Even though there are no constraints in this sketck, Synchronous Technology 2D finds geometric relations based on the curve(s) you select to edit and maintains these relations during*

change. Synchronous Technology 2D commands make it possible to make dynamic modifications to sketches while maintaining tangency constraints and connectivity.

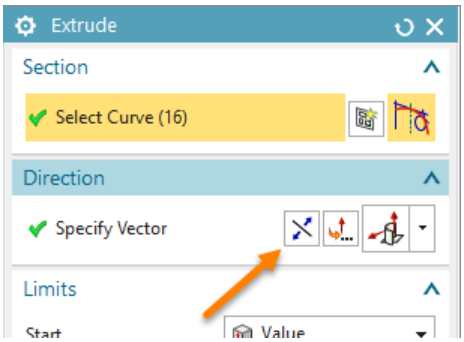
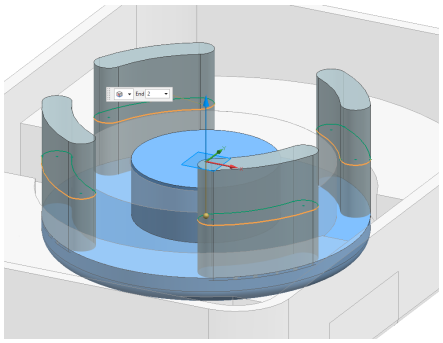
10	Choose **Move Curve** from the **Home** tab  Select the upper arc of the profile on the top → right side of the sketch. Click on the ball in the center of the translation widget (called **Orient Express**) → drag the center of the translation widget to snap it to the sketch origin Expand **Curve Finder** → go to the **Setting** tab and turn on **Select Symmetric**.	 

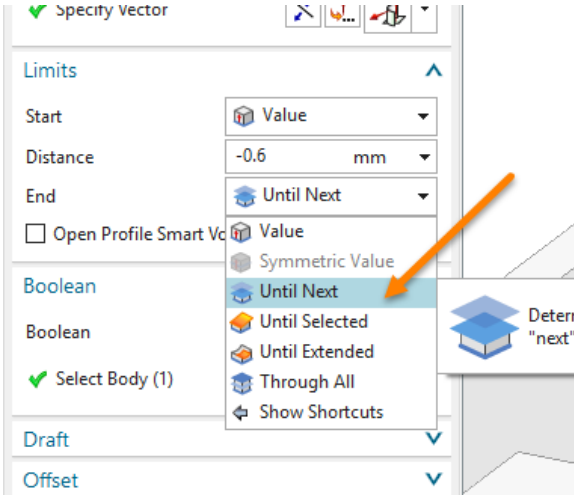
<table>
<tr>
<td></td>
<td>


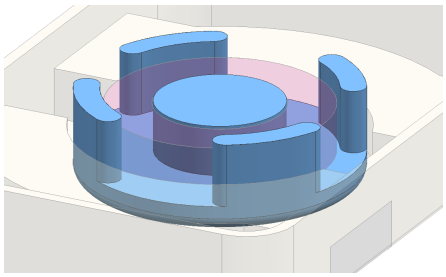
Enter 30 for the Angle → click Apply.

</td>
<td>

</td>
</tr>
<tr>
<td>11</td>
<td>While the Move Curve command is still active → select the lower arc → drag and snap Orient Express to the sketch origin (same as in the previous step) → enter 7 for the Angle → click OK.</td>
<td>


</td>
</tr>
<tr>
<td colspan="3">Note: Sketches modified with Synchronous Technology 2D are automatically updated according to changes in real-time. For instance, if you move or offset a curve, all associated geometry intelligently adapts. The relevant relationships are preserved while irrelevant relationships are removed to enable the intended change to be made with ease.</td>
</tr>
</table>

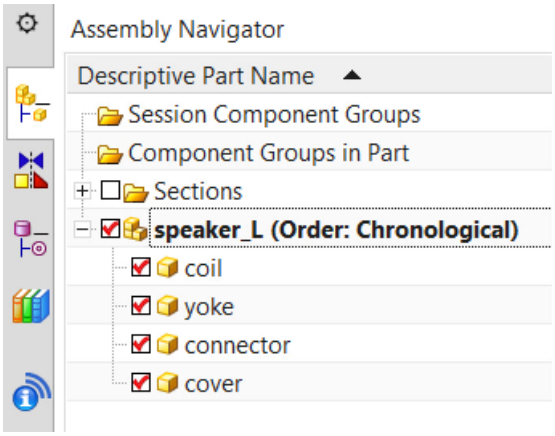
12	While the Sketch is still active → click **Extrude** on the **Home** tab Rotate the view so the preview of the extrusion is better seen The extrusion direction is chosen by default and should be along the positive Z axis. We want it to go in the opposite direction. Click on the **Reverse Direction** icon in the **Direction** group on the dialog.  Specify start and end limit for the extrusion. We will use a value for the start of the extrusion, and a trimming condition for the end. Start = **- 0.6** End = **Until Next** Set Boolean= **Unite** and click **OK**	



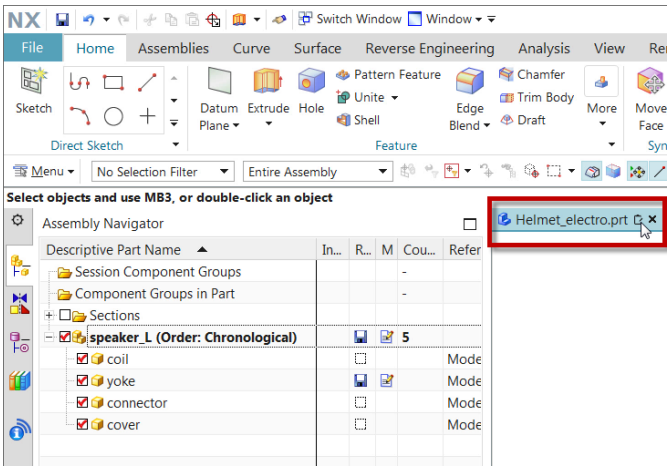


Design is completed

Double click speaker_L



Select Helmet_electro.prt Tab

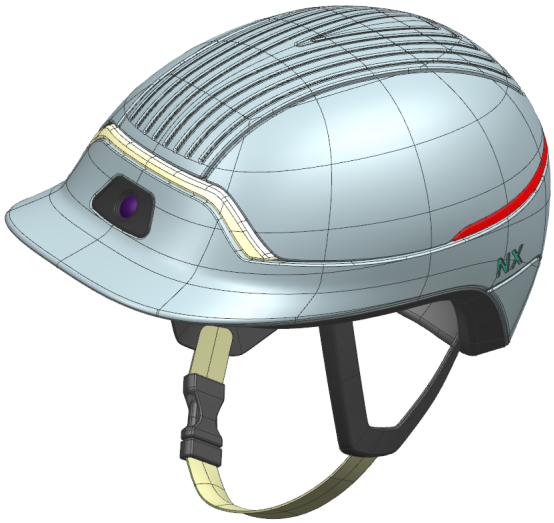
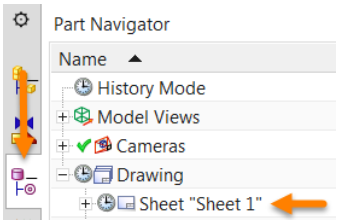
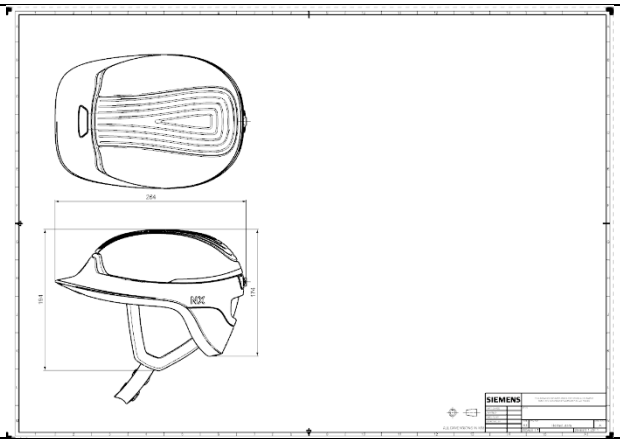
		
13	Go to **File → Close → All Parts** to return to the Tutorial Home page. Click "**No – Close**" in the **Close All Files** message window.	
End of activity		

Creating an assembly drawing with PMI in NX

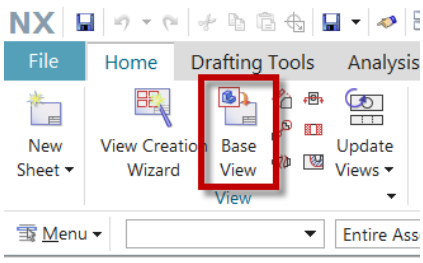
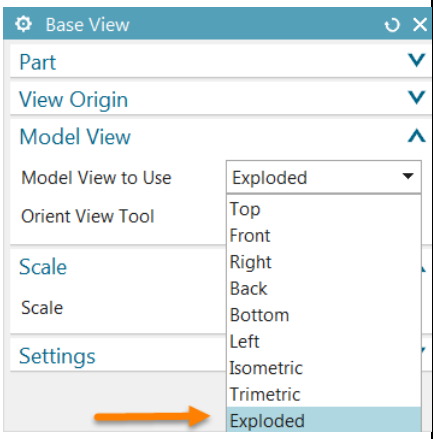
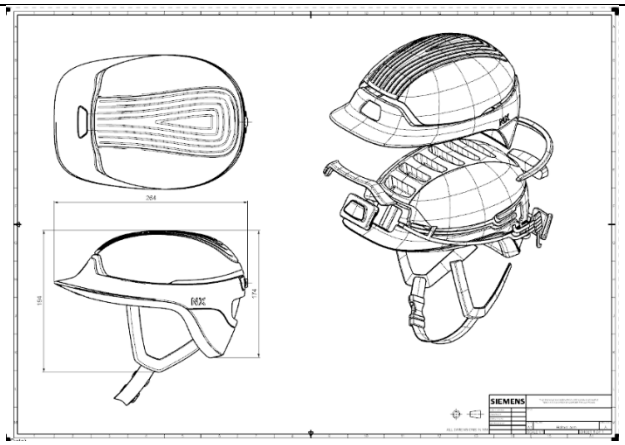
Introduction

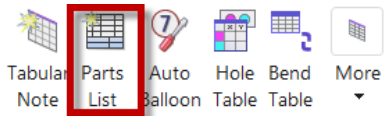
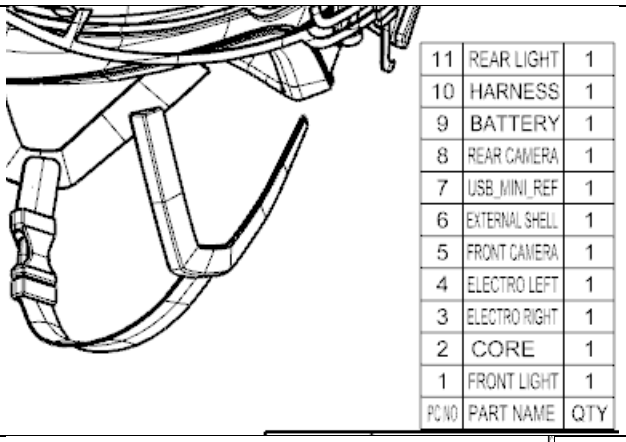
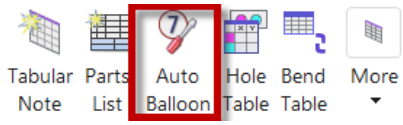
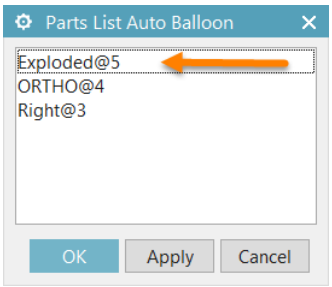
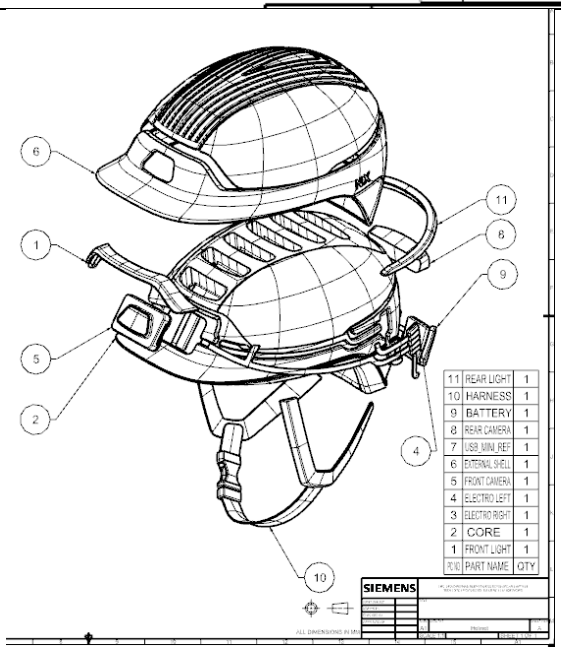
There is a growing need to document designs in 3D in support of the Model Based Definition (MBD) initiative many companies have or are undertaking. Working with 2D legacy data involves significant manual the effort of recreating information from a drawing onto the 3D model. To simplify and automate these workflows, NX has tools such as 'Convert to PMI' to automatically convert drawing views and objects to model views and PMI objects.

This tutorial walks you through the process of creating an assembly drawing sheet, adding exploded view to the drawing and generating a parts list for the drawing sheet. We will then add GD&T symbols to the drawing and automatically generate PMI using the 'Convert to PMI' feature.

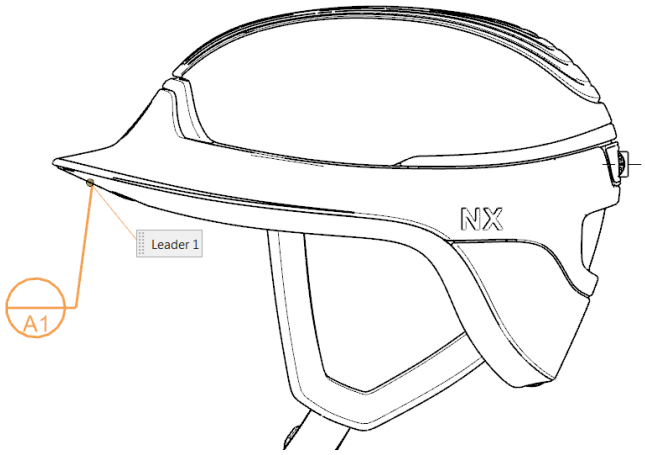
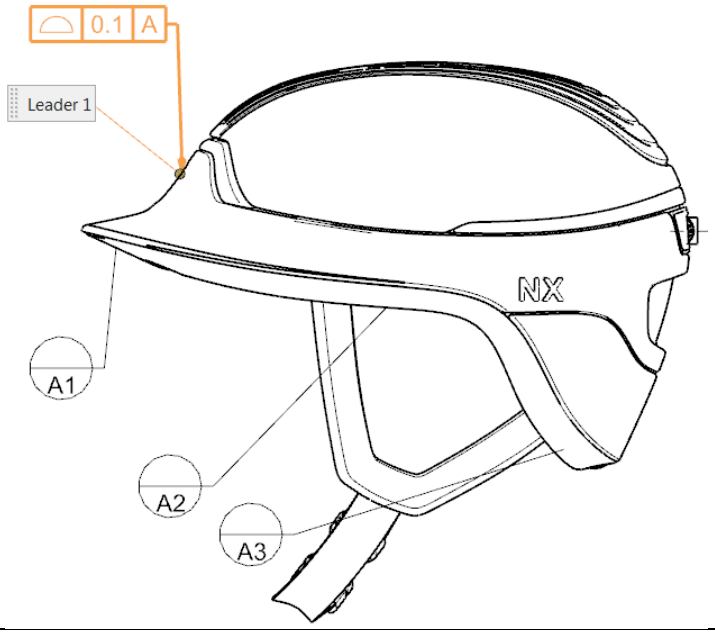
1	The model used for this tutorial is 'Helmet asm.prt'	
	***Note**: We have already created a drawing and placed a couple of views in order to accelerate the workflow for the purposes of this test drive. This review does not use NX "Master Mondel Concept" where drawing is in a separate file from the model (or assembly file).*	
2	In the **Part Navigator** → expand **Drawing** → double-click "Sheet 1" to open it  There are two existing views.	

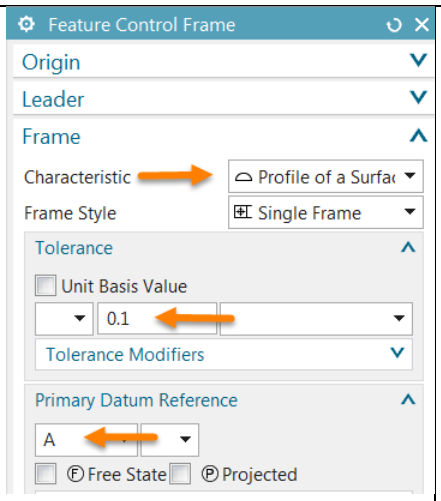
***Note**: The objective of the next few steps is to demonstrate how you can add an exploded assembly view to a drawing and automatically create baloons and the corresponding table, and make manual adjustments if necessary.*

3	**Home** tab → click **Base View**  Under **Model View to Use** → set **Model View to Use** to **Exploded**  Place it on the right side of the sheet next to the two existing views Click **Close**	

4	**Home** tab → click **Parts List**. Place it next to the exploded view.  	
5	**Home** tab → click **Auto Balloon**  Select the parts-list → click **OK** **Parts List Auto Balloon** window → select the exploded view → click **OK**  The position of the balloons can be changed by dragging them.	

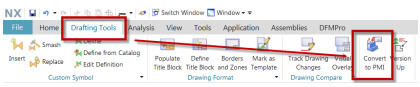
***Note**: The objective of the next few steps is to add annotations to a drawing. Although these are just a few basic types, you will be able to get a feel for the UI and interaction in performing this task.*

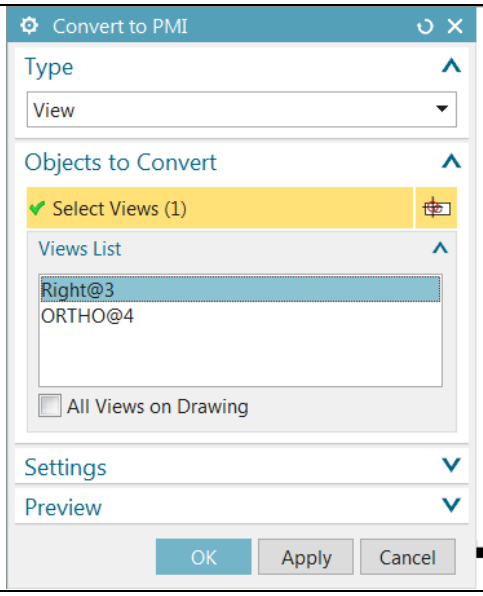
6	**Home** tab → click **Datum Target** Click and hold on a point towards the front of the helmet. Drag to an appropriate location and place it. Add two more datums along the same curve; one in the middle and the second towards the bottom. (See the next image in Step 7). Click **Close**	
7	**Home** tab → click **Feature Control Frame** Expand **Frame**, set the **Characteristic** to **Profile of a Surface**. Under **Tolerance**, enter a value **of 0.1** and set the **Primary Datum Reference** to **A.**	

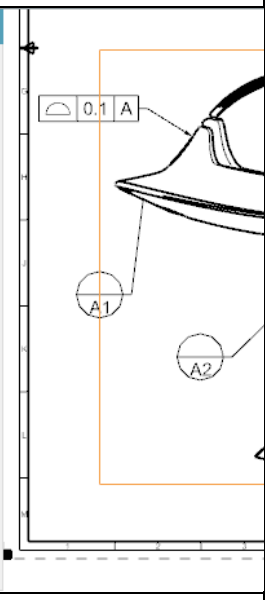
<table>
<tr>
<td></td>
<td> 

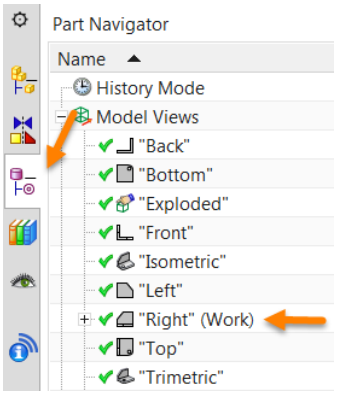
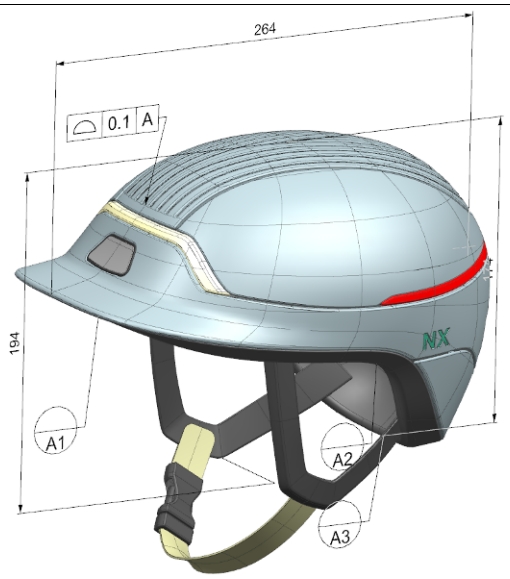
Click and hold on a point and drag to an appropriate location and place it.

Click Close</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>Drafting Tool tab → click Convert to PMI

Under Type select View → From the Views List → select “ORTHO@6” → click OK.</td>
<td> 


</td>
</tr>
<tr>
<td colspan="3">Note: Lastly, we can now convert the annotation in a drawing view to PMI objects in 3D. You have options to convert a single view, multiple views, or the entire drawing including multiple sheets.</td>
</tr>
</table>

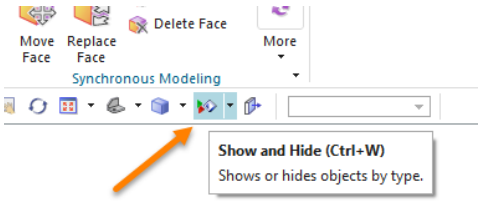
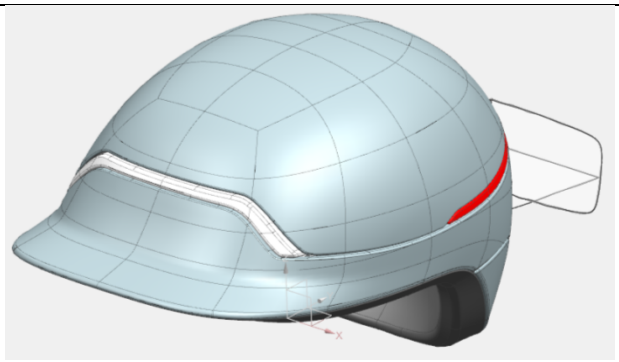
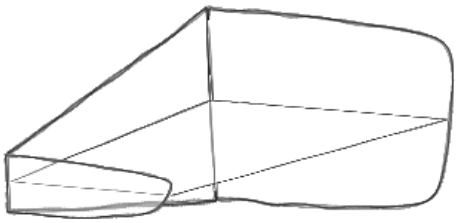
9	On the **Application** tab, click **Modeling**. In the **Part Navigator**, expand **Model Views** and double-click "Right".  The PMI objects added to the drawing sheet are now associated with the 3D model making it easier to access critical product information and enabling a model-based definition workflow.	
10	Go to **File** → **Close** → **All Parts** to return to the Tutorial Home page. Click "**No – Close**" in the **Close All Files** message window.	
End of activity		

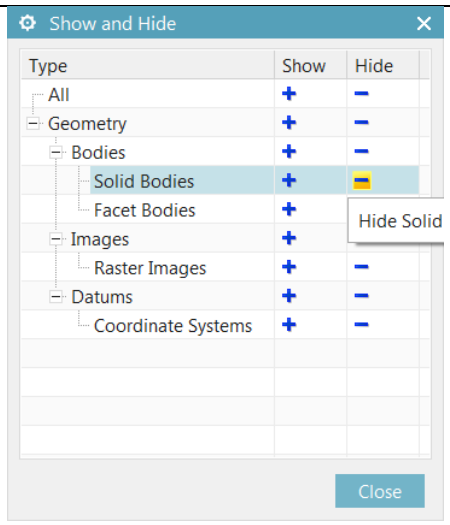
Creating an aesthetic shape using NX Realize Shape (Free Form Tool) in NX

Introduction

Fairly often the design of a product calls for creation of a more freeform shape. Conventional tools require a high level of expertise and still can take more than desired time to create. NX offers a product called Realize Shape that is based on the subdivision technique which enables designers to create complex shapes in a fraction of the time compared to conventional tools. Equally important is the ability to use all NX commands on the resultant geometry in order to add the required details that tend to be more mechanical and require traditional construction methods and tools.

NX Realize Shape provides an intuitive approach to sculpt 3D solid or sheet bodies by manipulating and subdividing a control cage. In this tutorial, we begin by using a raster image of a conceptual sketch to create a primitive shape that is transformed into an organic shape. We will then use this shape to modify the helmet shell and finally, add a logo to the helmet.

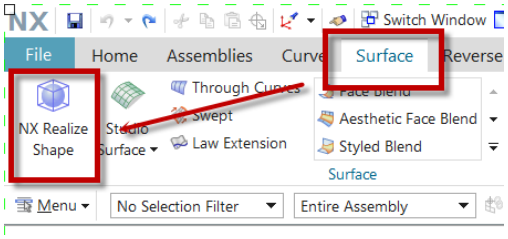
1	The model used for this tutorial is "Helmet_SubD_Replaceface_Emboss.prt" **Top Border** bar → click **Show and Hide** → Click "**–**" under **Hide** for **Solid Bodies** → Click **Close**. 	 



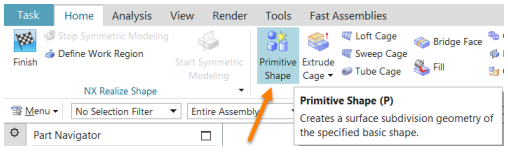
This would display just the raster image to be used for creating the shape.

***Note**: The design typically starts with a primitive shape. NX offers a number of shapes like a sphere, a block, a cylinder, and others. For this activity a block is the most suitable shape to start with.*

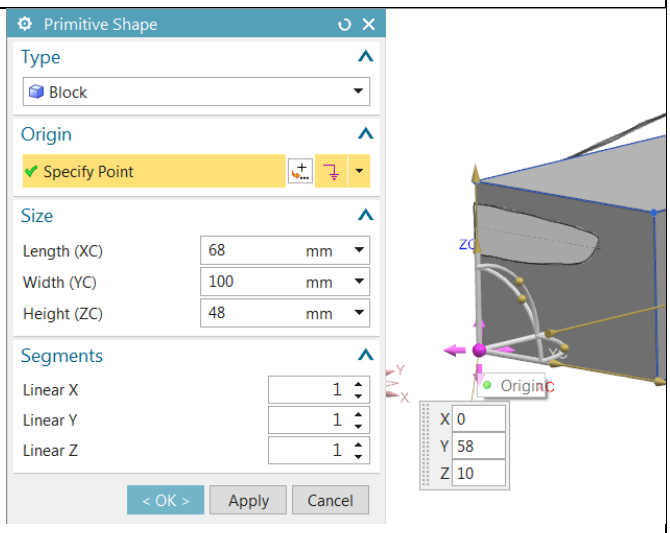
2 **Surface** tab → click **NX Realize Shape**.



On the **Home** tab → click **Primitive Shape**

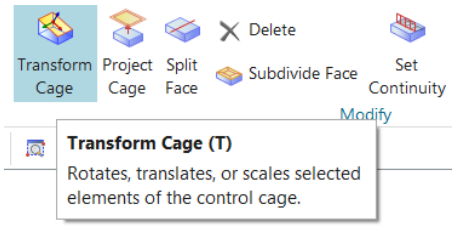
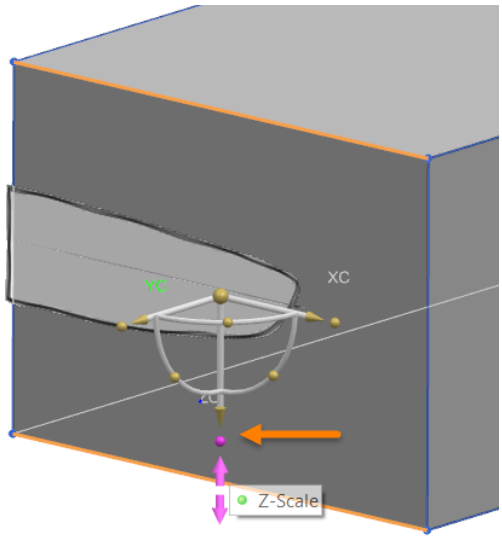


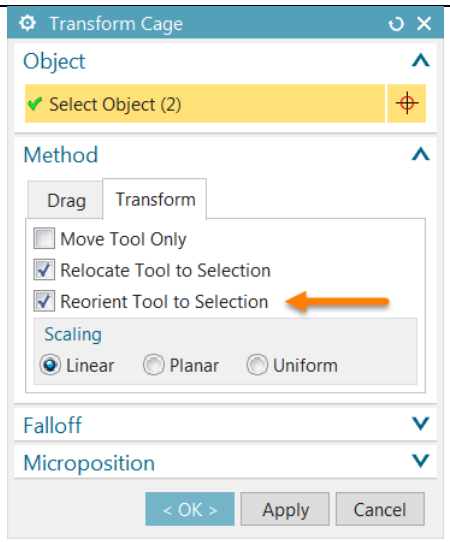
Set the **Type** as **Block** → Enter these values for the **Size**:

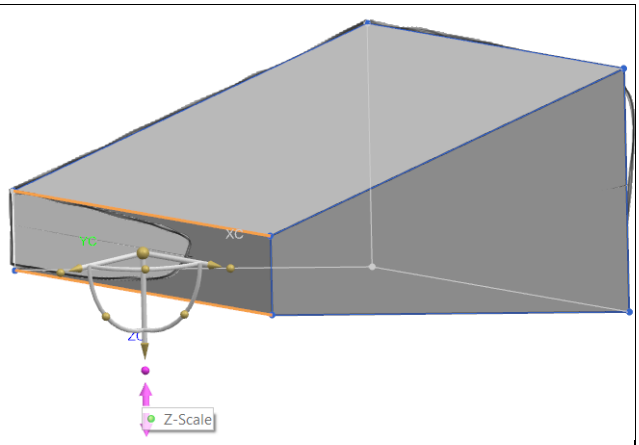


	Length = **68** mm **Width** = **100** mm **Height** = **48** mm Click on the origin in the graphics window and enter these values: **X** = **0** **Y** = **58** **Z** = **10** Click **OK**	

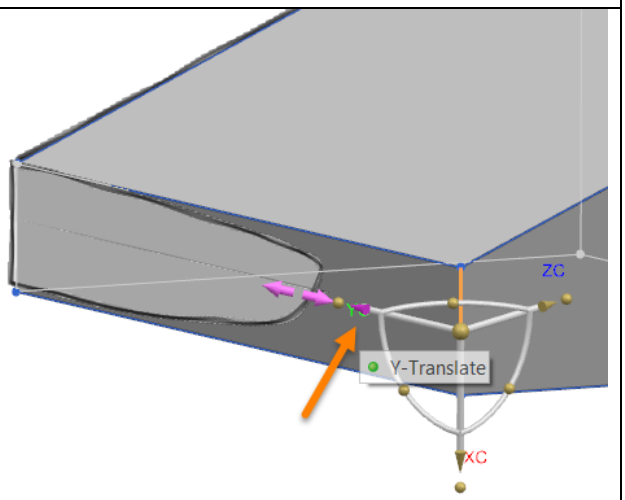
***Note**: A bicycle helmet typically has a freeform and organic shape. The cutouts at the back of the helmet were designed to match the theme of the overall design, and the shape of the tool required to make these cutouts is also freeform. This is a a good example of how Realize Shape can be effectively used to quickly create a somewhat complex shape. Using conventional tools for modeling this shape will take significantly longer and may requie advanced surfacing skills. Realize Shape creates high quality geometry with natural continuity between faces.*

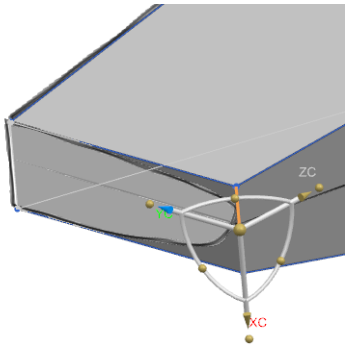
3	On the **Home** tab → click **Transform Cage**  Under **Method** → click on the **Transform** tab → check **Reorient Tool to Selection** Select the two parallel edges along X axis and drag them using the sphere to align with the raster image.	

<table>
<tr>
<td></td>
<td>


Click Apply.
</td>
<td>


</td>
</tr>
<tr>
<td>4</td>
<td>Select the vertical edge → drag it towards the raster image using the Y Translate grab handle.

Click OK → to close the Transform Cage window.</td>
<td>




</td>
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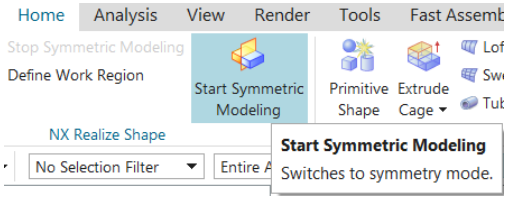
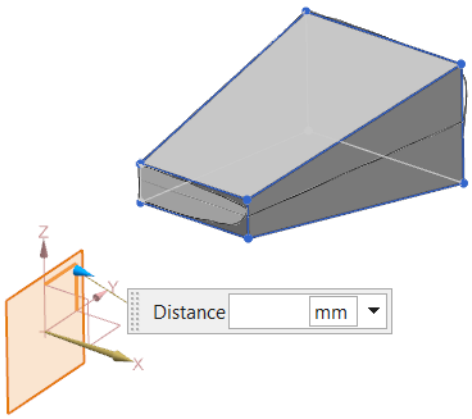
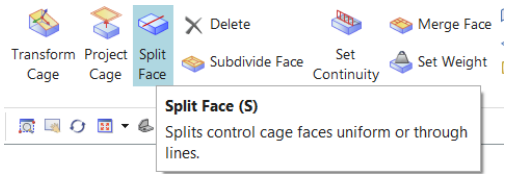
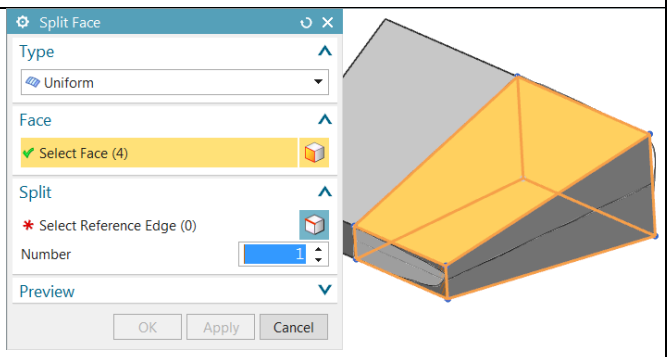
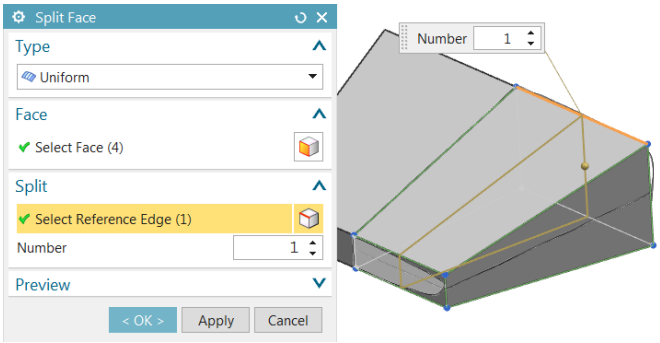
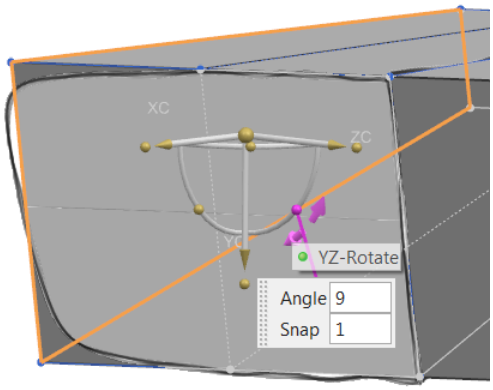
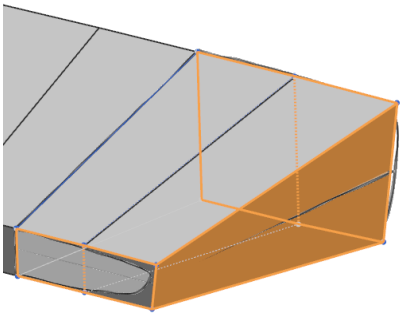
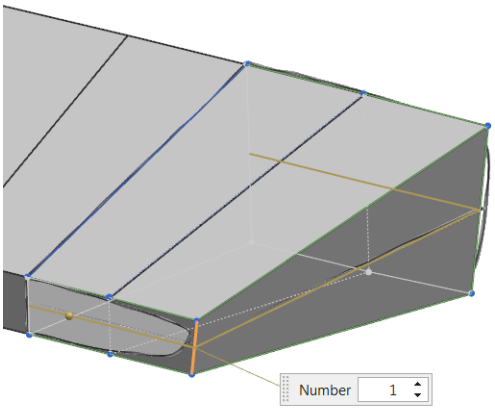
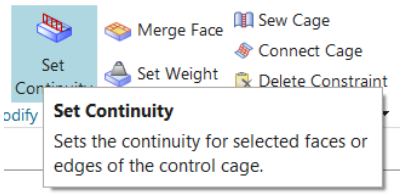
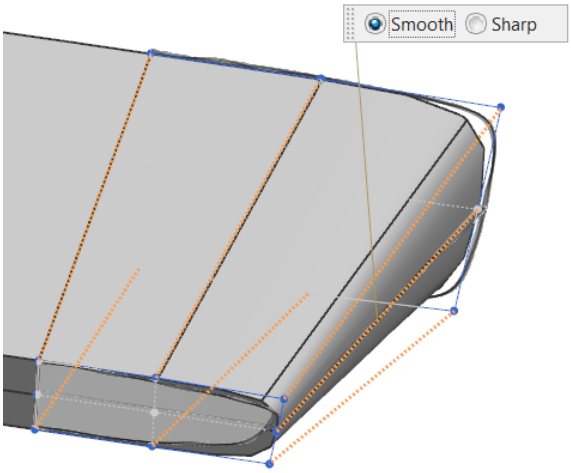
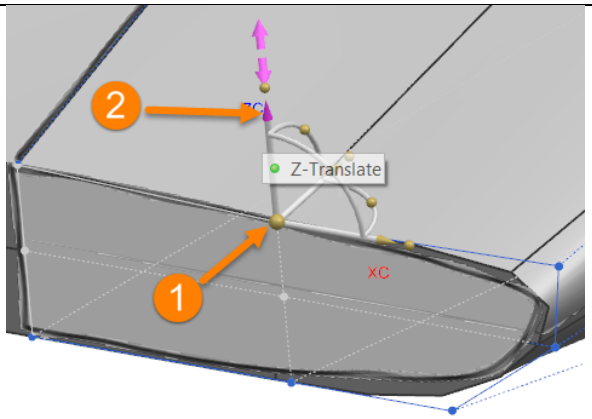
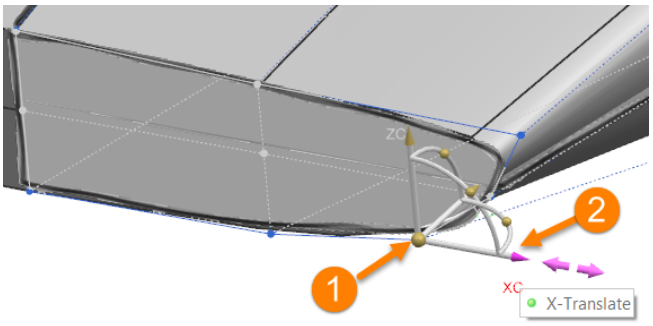
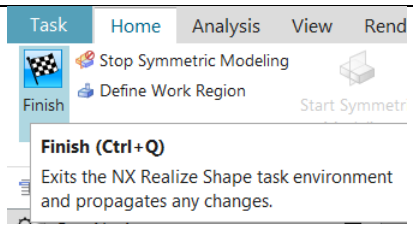
</table>

5	**Home** tab → click **Start Symmetric Modeling**.  Select the **Y-Z** plane and click **OK**.	
***Note**: In order to to achieve the desired shape, we split the faces of the original primitive shape. This gives us the ablility to edit the shape locally and add more details.*		
6	**Home** tab → click **Split Face**  Select the four highlighted faces (front, back, top, and bottom). Click on **Select Reference Edge** → select the horizontal edge on the top of the back face to split the four faces Click **OK**.	 

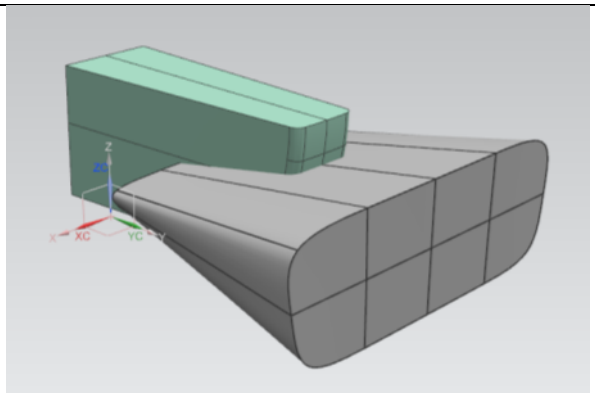
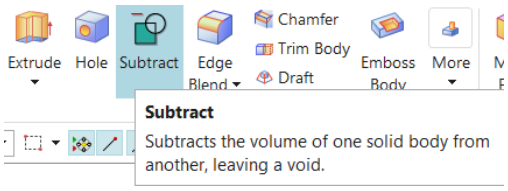
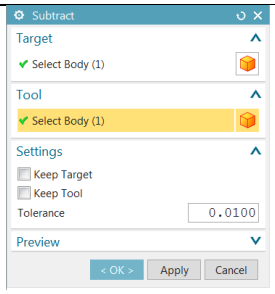
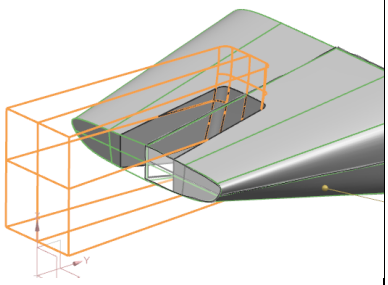
7	**Home** tab → click **Transform Cage** Select outside face to transform → click on the **YZ Rotate** handle → Enter **Angle** = **9** (if the rotation moves in the wrong direction, change value to -9) Drag outside face to match the image Click **OK**	
8	**Home** tab → click **Split Face** Select the five faces (two each in the front and back, one on the side) to split and the vertical edge as the reference. Click **OK**	 

9	**Home** tab → click **Set Continuity**  Select the 7 horizontal edges in the Y direction → set **Continuity** to **Smooth** and click **OK**	
10	**Home** tab → click **Transform Cage** Under **Method** → uncheck **Reorient Tool to Selection**. Select the vertex → click the **Z Translate** drag handle to transform the cage to match the raster image profile. Click **Apply**. Repeat the procedure for the other vertices of the cage. Once the shape matches the raster image profile, click **OK**. Click **Finish**.	 

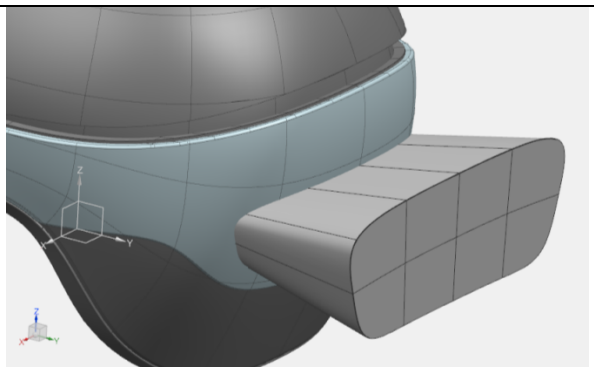
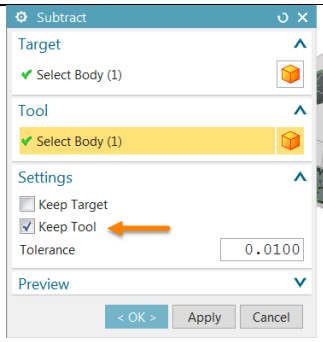
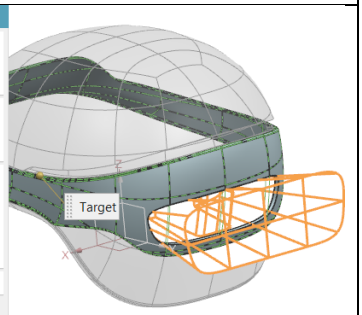
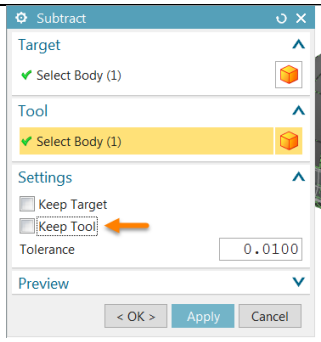
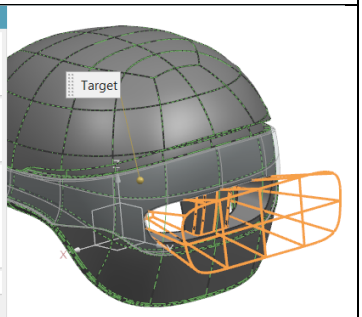
		

***Note**: Now that we have finished designing the initial shape of the tool which we will use to make the cutouts in the back of the helmet, we can use more conventional design tools to finish the required model. Since subdivision modeling is part of the core modeling environment in NX, you are able to seamlessly use all of the modeling tools in NX to add to the subdivision body.*

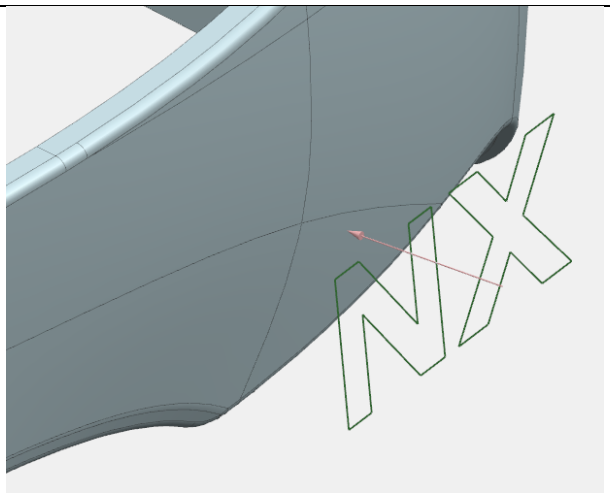
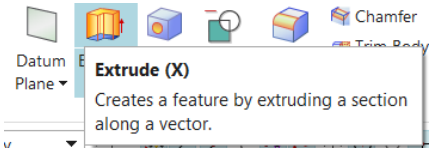
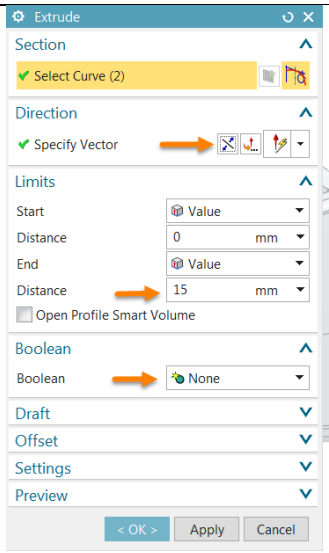
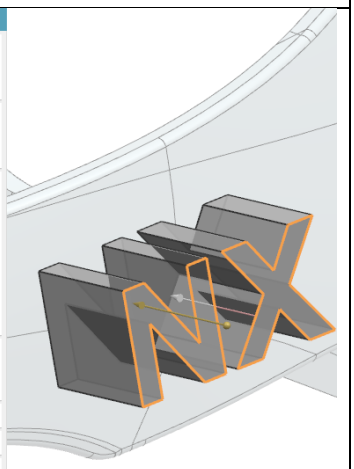
For the purposes of this test drive we have already created the second subdivision model which we will now use to remove the required material from the body we just designed.

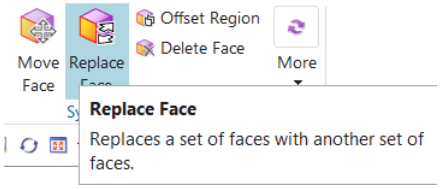
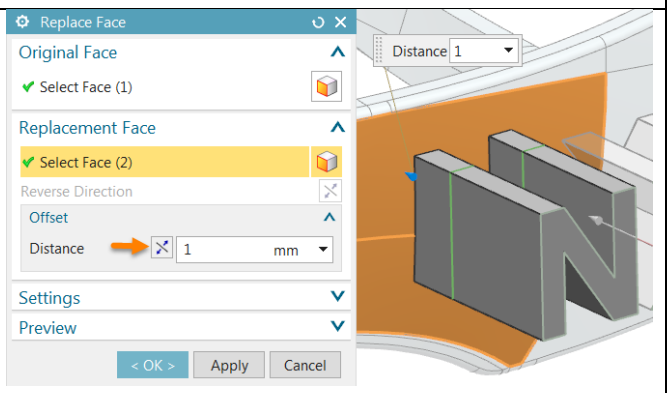
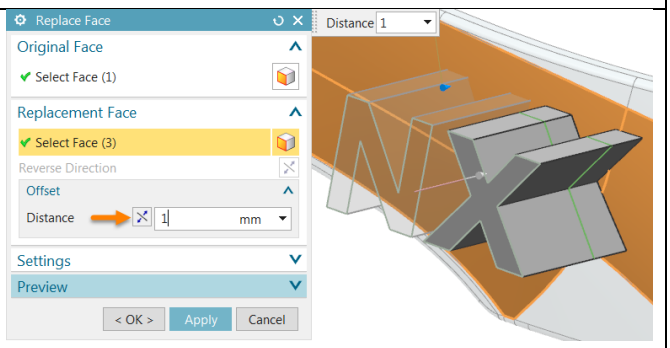
11	Go to the **Top Border** bar → click **Show and Hide** → Click – under **Hide** for **Raster Images**. Click **Close**. In the **Part Navigator** → right-click on **Subdivision Body (10)** → select **Show**.	
13	Go to the **Home** tab → **Feature** group → **Subtract**.  Select **Subdivision Body (13)** as the **Target** → **Subdivision Body (10)** as the **Tool**. Click **OK**.	 

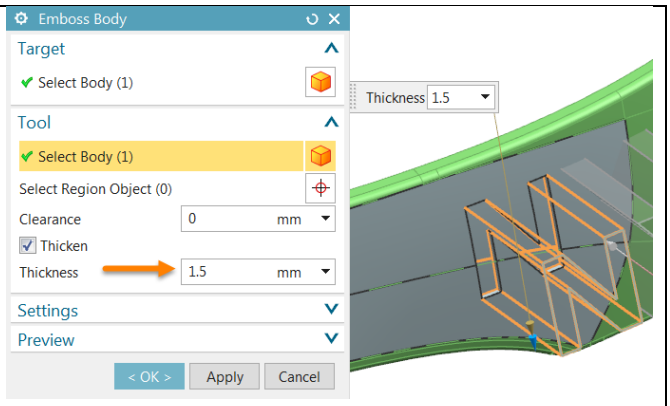
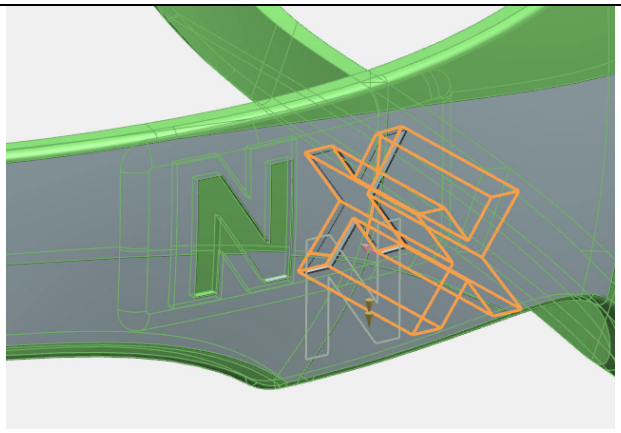
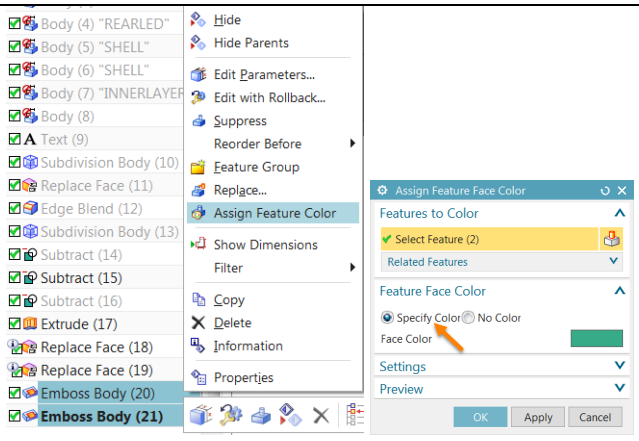
***Note**: In the next few steps we will use the tool body we made in order to create the required cutout in a couple of parts in the back of the helmet.*

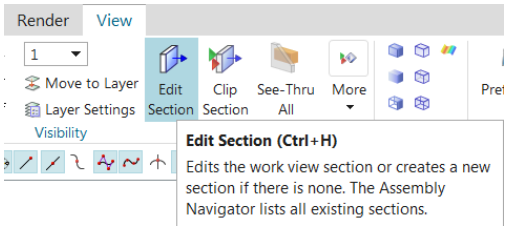
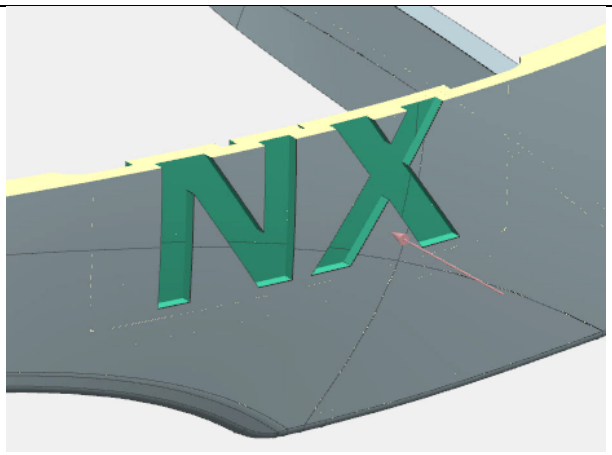
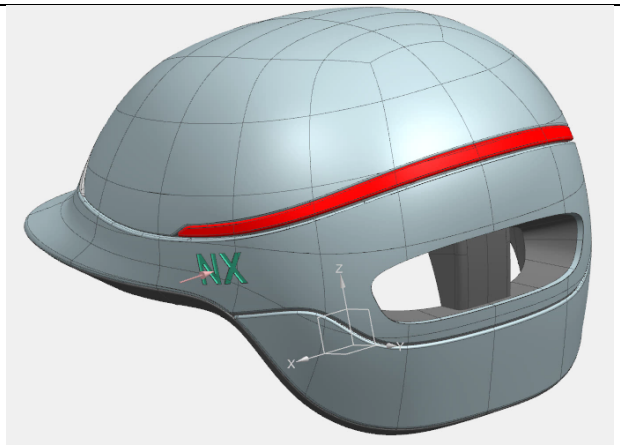
14	In the **Part Navigator** → right-click on **Body (2) "ENGRAVING"** → select **Show**. Similarly → right-click on **Body (7) "INNERLAYR "**→ select **Show**.	
15	Go to the **Home** tab → **Feature** group → **Subtract**. Select **Body (2) "ENGRAVING"** as the **Target** → **Subdivision Body (13)** as the **Tool**. Under **Settings** → check **Keep Tool**. Click **Apply**.	 
16	In the Subtract window, select **Body (7) "INNERLAYR "**as the **Target** → **Subdivision Body (13)** as the **Tool**. Under **Settings** → uncheck **Keep Tool**. Click **OK**.	 

***Note**: In the next steps you will be able to experience how NX can help solve another interesting design challenge. Many times a design will call for an emossed or indented shape. We took an NX font and converted it to curves, which is one of the commands in NX. Now in a few simple steps we will use these curves to create a solid body and use it to emboss the helmet geometry.*

17	In the **Part Navigator** → right-click on **Body (7) "INNERLAYR"** → select **Hide**. Right-click on **Text (9)** → select **Show**.	
18	On the **Home** tab, select **Extrude** from the **Feature** group.  For **Select Curve** → click on the text. Under **Direction**, click **Reverse Direction** . Enter **Start** = **0** → **End** = **15** and **Boolean** = **None** Click **OK**.	 

19	On the **Home** tab, **Synchronous Modeling** group, select **Replace Face**.  For **Select Face** → select the face of letter N closer to the helmet For **Replacement Face** → select the two faces of the helmet. For **Offset** → click on **Reverse Direction** (blue arrow pointing towards the helmet) and enter **Distance** = **1**. Click **Apply**.	
20	Repeat the procedure in the previous step for letter X and select the three faces. Click **OK**.	

21	On the **Home** tab → click **Emboss Body**. Select **Body (2) "ENGRAVING"** as the **Target** and **"N"** as the **Tool**. Enter **Thickness** = **1.5** Click **Apply**.	
22	Repeat the procedure in the previous step for letter X → change **Thickness** = 1.6 Click **OK**.	
23	In the **Part Navigator** → right-click on **Text (9)** → select **Hide**. In the **Part Navigator** → Ctrl select the two **Emboss Body** features in the tree and go to **Assign Feature Color**. Under **Feature Face Color**, click on **Specify Color** and select a color.	

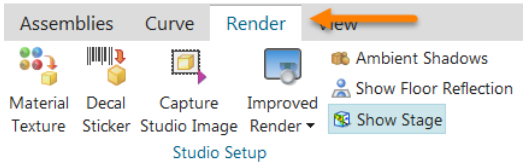
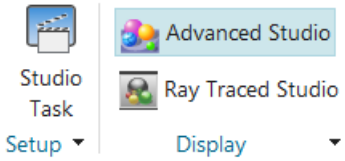
24	On the **View** tab → click on **Edit Section** to verify that the depth of the emboss is uniform.  Click **Cancel**.	
25	Go to the **Top Border** bar → click **Show and Hide** → Click on **+** under **Show** for **Solid Bodies**. Click **Close**.	
26	Go to **File** → **Close** → **All Parts** to return to the Tutorial Home page → Click "**No – Close**" in the **Close All Files** message window.	

End of activity

Rendering and Visualization using NX Ray Traced Studio

Introduction

Photorealistic rendering allows you to make decisions about aesthetics, function, and materials of a product early in the design process. The Iray based rendering engine in NX delivers progressive ray tracing in a fully integrated environment with Ray Traced Studio. It has an extensive library of materials, textures and scenes. In this tutorial, we will practice applying various materials to the parts in the assembly and changing the background scene to review the lighting design.

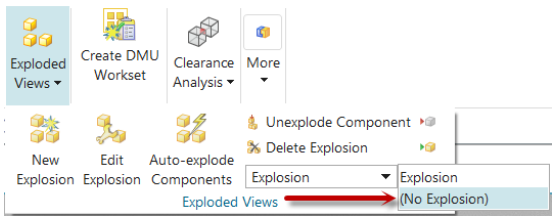
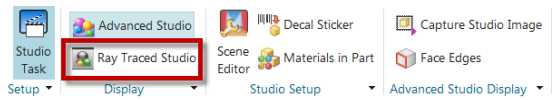
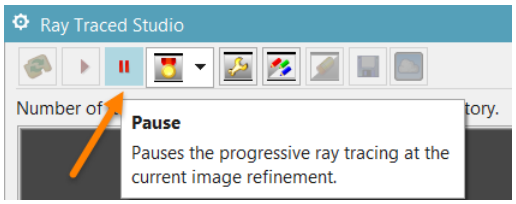
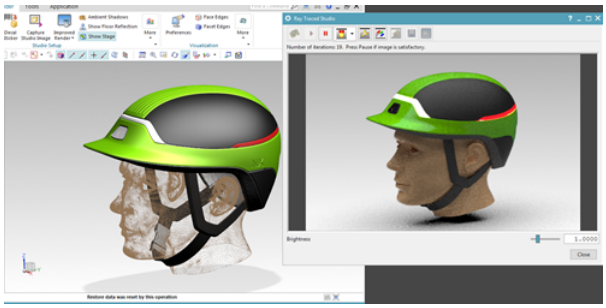
1	The model used for this tutorial is "HelmetAssembly2.prt " Click on the **Render** tab.  An exploded view of the model is saved with materials and scene already applied in **Advanced Studio** display.  	

***Note**: NX offers three rendering modes to suit the various product development stages such as design evaluation, design reviews, generating high-quality images for design communication.*

***True Shading**: A simple environment with basic materials, scene lights, reflections, and backgrounds for quick design reviews and capturing images of various states of design.*

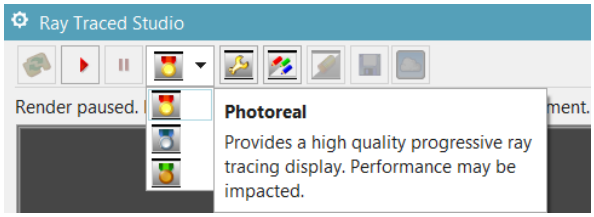
***Advanced Studio**: A nearly photo-realistic display, with more sophisticated materials, image-based lighting, and shadows.*

***Ray Traced Studio**: A ray traced image in real-time, that displays inter-object reflection, refraction, and global illumination effects in a separate window with dynamic updates. Capable of generating high-quality renderings for effective design communication.*

2	**Assemblies Tab → Exploded Views** dropdown → select No Explosion 	
3	On the **Render** tab → select **Ray Traced Studio**  The rendering starts as soon as the **Ray Traced Studio** window is displayed. Pause the rendering in the **Ray Traced Studio** window after a few iterations.  Tile the two windows such that the model and the rendering can be visualized side-by-side.	

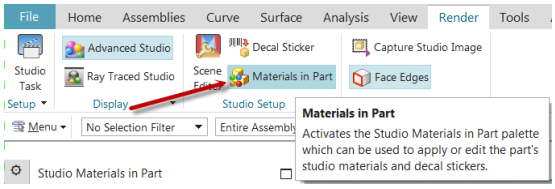
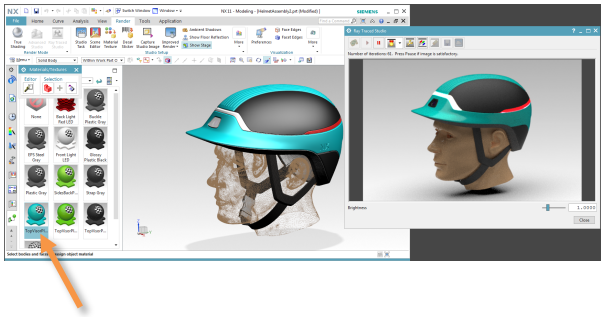
***Note**: The Ray Traced Studio window menu gives you the ability to pause and restart the rendering process until you are satisfied with the rendered result. It has three render modes that lets you choose the level of interactivity and performance.*

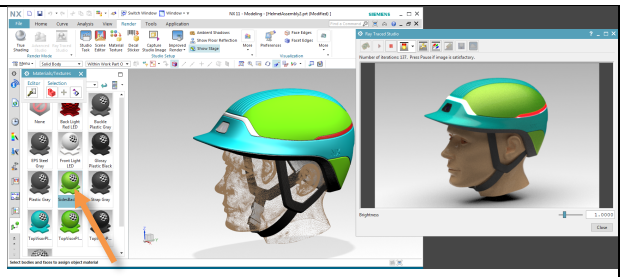
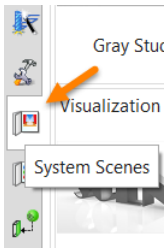
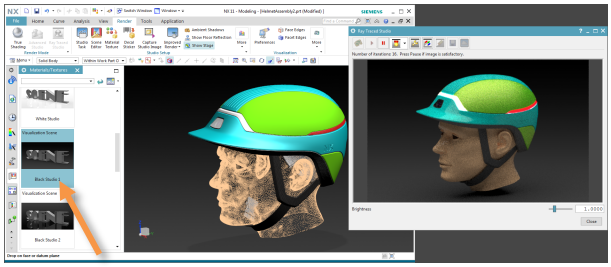
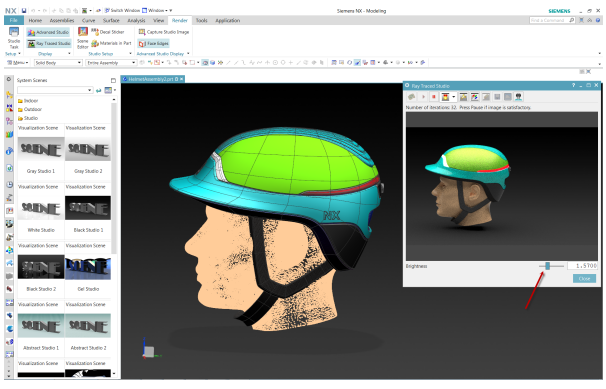
Photoreal: *Uses a physically-based, path-tracing render mode that generates photorealistic imagery with full global illumination effects.*


Ray Traced Studio
Render paused.
Photoreal
Provides a high quality progressive ray tracing display. Performance may be impacted.

Quality Interactive: *Uses an interactive raytracing render mode optimized to give the best approximation of* ***Photoreal*** *with increased interactivity.*

Fast Interactive: *Uses an interactive raytracing render mode optimized for the fastest view dynamic performance.*

4	On the **Render** tab → click **Materials in Part**  The **Studio Materials in Part** are displayed. **Drag/drop TopVisorPlasticBlue (Coated)** from the materials palette to the neon green Visor in the graphics window. Repeat the procedure for the Top panels of the outer helmet shell. Click **Start/Resume** in the **Ray Traced Studio** window.	
5	**Drag/drop SidesBackPlasticPearlescentNeonGreen** to	

	the black Side and Back panels of the helmet shell	
*Note: The **Studio Materials in Part** palette shows the materials that have been used in the model. NX comes with a broad, pre-defined set of materials that can be accessed from **System Studio Materials**. These are customizable materials based on NVIDIA Material Definition Language (MDL) that support advanced material characteristics such as light emission.*		
6	Go to **System Scenes** on the Resource Bar.  In the **Studio** folder, select **Black Studio Scene 1.**	
***Note**: **System Scenes** in NX consist of indoor and outdoor scenes that have pre-defined background, stage, reflection, shadow, and global illumination attributes. Applying a visualization scene to your model to puts it into visual context. Existing **System Scenes** can be edited to make further changes to your background, scenes, lighting, and reflection, if necessary.*		
7	Adjust the **Brightness** using the slider located at the bottom of the **Ray Traced Studio** window. Drag the **Brightness** slider to the right to brighten the scene. Click **Pause** after a few iterations in the **Ray Traced Studio** window.	

8	Go to **File** → **Close** → **All Parts** to return to the Tutorial Home page. Click "**No – Close**" in the **Close All Files** message window.	

End of activity